



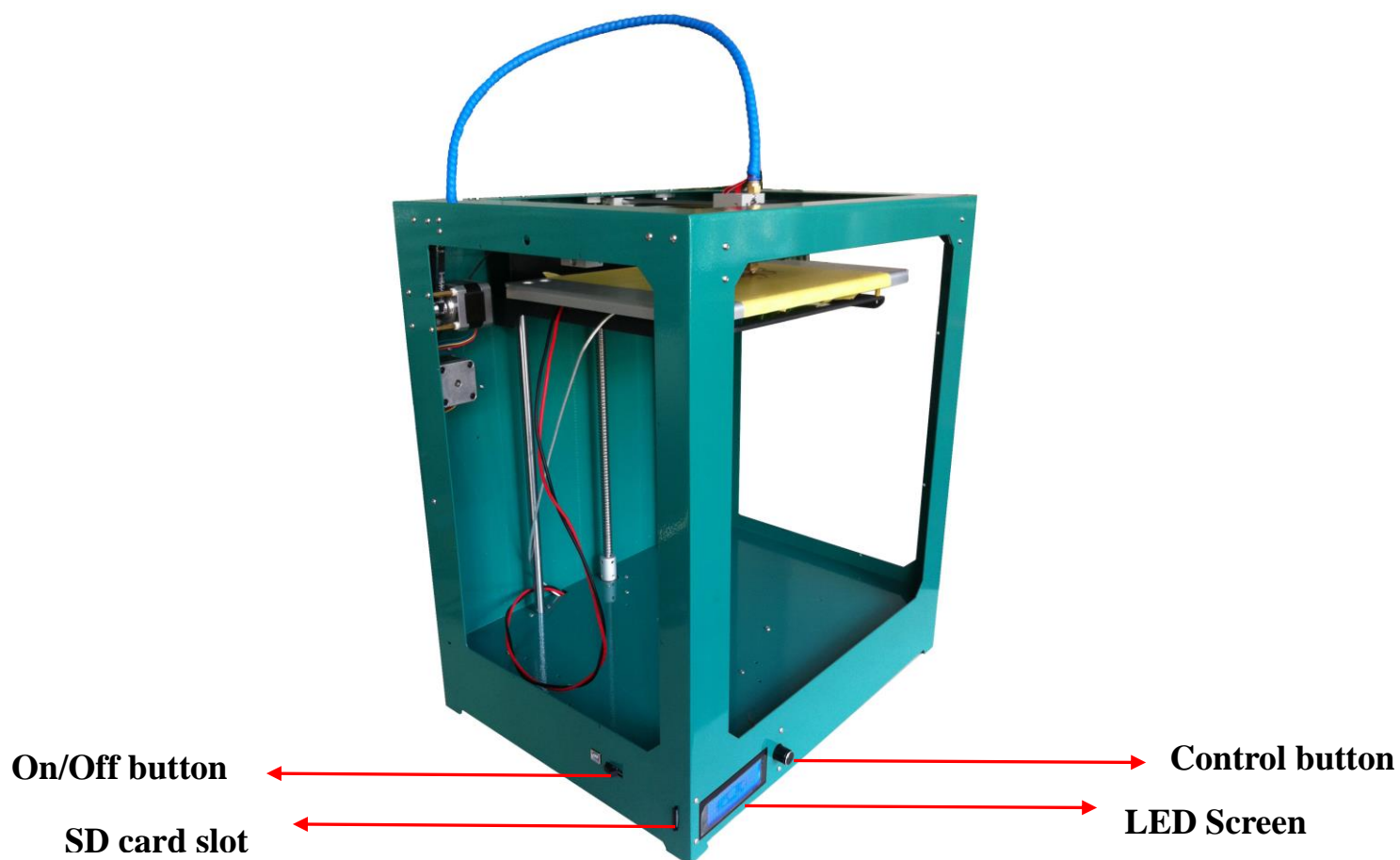
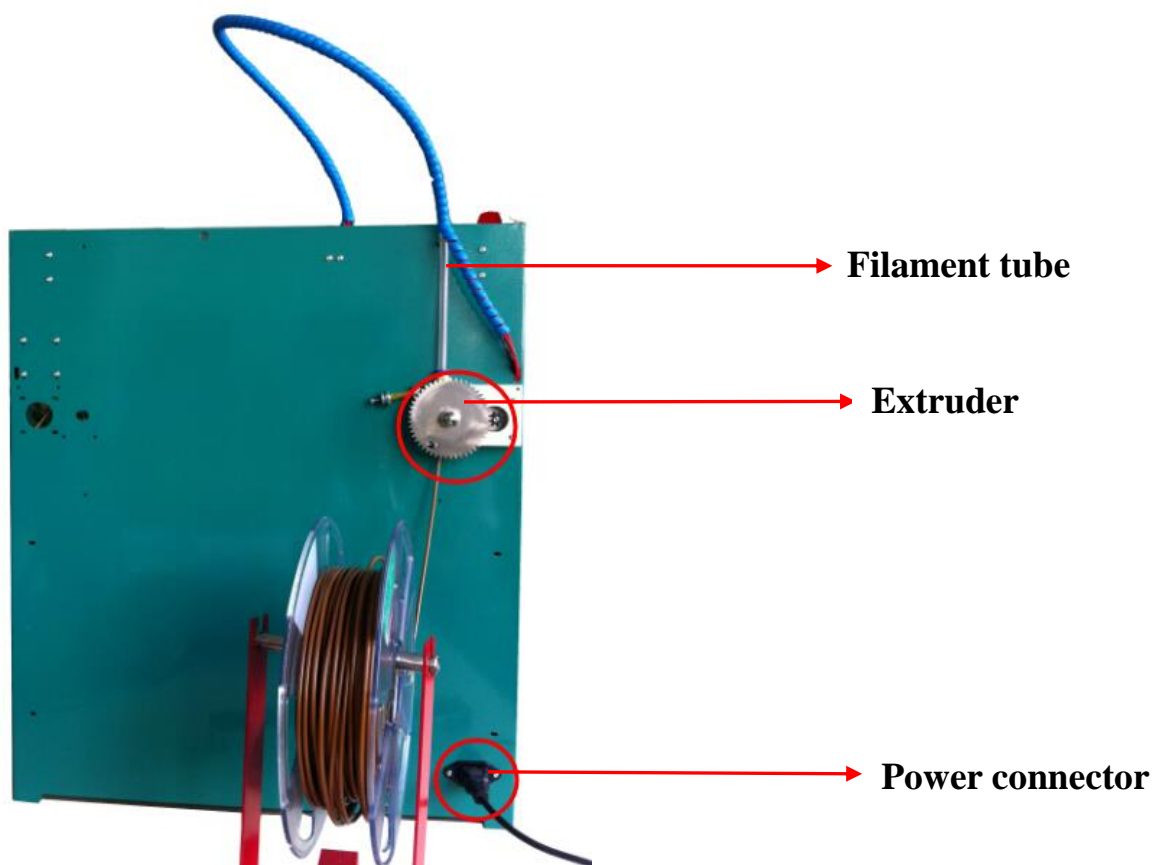
Focus3D Printer Setup Manual

Part I: Introduction to 3D Printer	2
Part II: A Few Easy Steps to Set Up 3D Printer	4
Part III: Install the Software	6
Part IV: Run the Software	10
Part V: 3D Print Model Parameters Setting	14
1-> Print Model Parameters Configuration	14
1.1> Switch to Full Setting	14
1.2.1 Basic Configuration	15
1.2.2 Advanced Configuration	16
1.2.3 Expert Configuration	17
1.2.4 Plugins Settings	18
1.2.5 Machine Settings	18
Part VI: Start to Print	19
Part VII: Change the Filament	21



Part I: Introduction to Focus3D Printer

Technical Parameters	
Print Technology	FDM
Layer Resolution	0.06-0.2mm
Layer Thickness	0.06-0.5mm
Nozzle Diameter	0.4 mm
Extruder Temp	0-240°C
Hot Bed Temp	0-110°C
Connectivity	SD Card
Print File Format	Input:STL/OBJ , Output:G/X3G
Operating System	Windows XP/7/Vista, Linux, Mac
Software	Focus3D Cura
Print Speed	10-150mm/s
Input voltage	AC100-250V, 50/60H,250W
Printing Material	PLA/ABS
Filament Color	White/Black/Yellow/Red/Blue/Green
Filament Diameter	3.0 mm



Part II: 3D Printer Setup

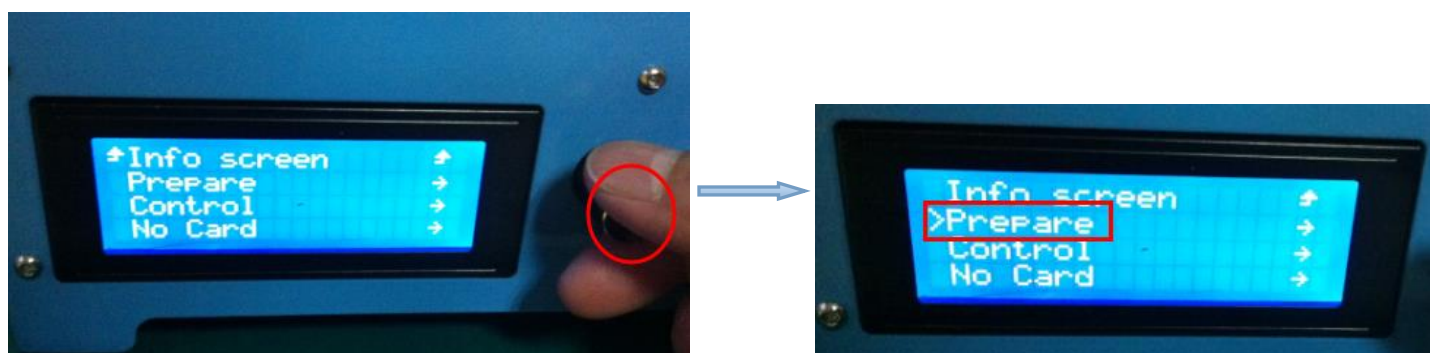
As our 3D printer has been fully adjusted and calibrated before it is shipped, you just need a few easy steps to finish installation as following:

1-> Connect the 3D Printer to power supply with the power adapter

2-> Press the power button

3-> Click the control button, rotate it and select “Prepare”, click and select “Preheat PLA” or “Preheat ABS”.

3.1> Rotate the button and select “Prepare”

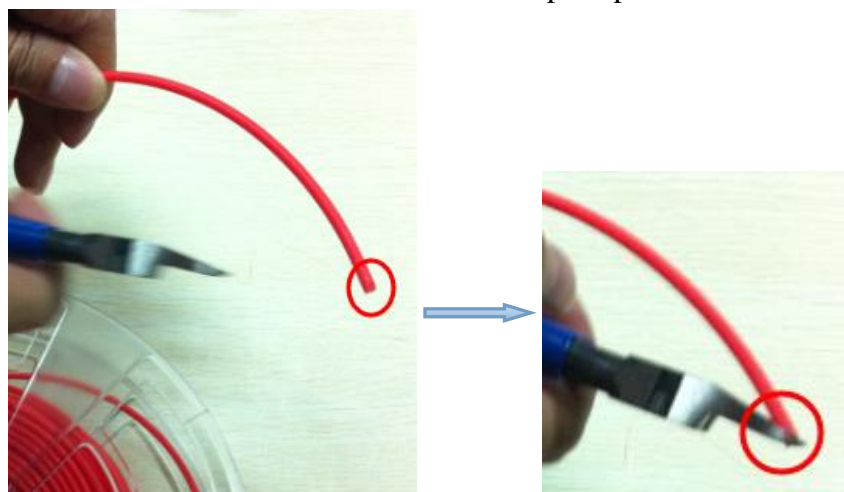


3.2>Select Preheat PLA or Preheat ABS



4-> Manually feed the filament into the filament tube and rotate the feeding gear to loose it, let the filament come out smoothly. When you see filament flow out smoothly, please rotate the feeding gear to chuck it.

4.1> Trim one end of the filament to a cusp shape

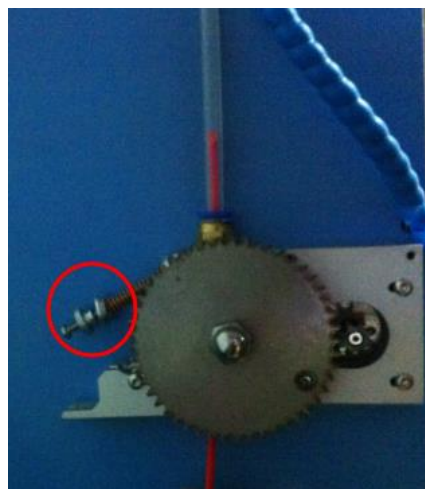


4.2> Begin to feed the filament

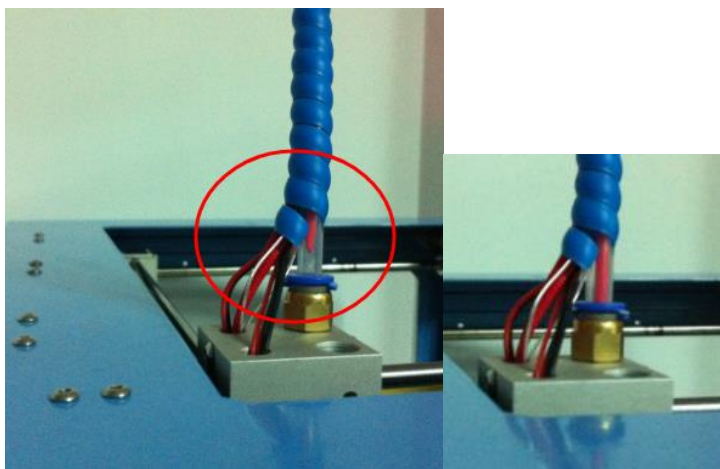
Rotate the feeding gear and loose it



Begin to feed the filament



See the filament and push it to flow out from the nozzle



The filament flow out smoothly



Stuck the filament gear



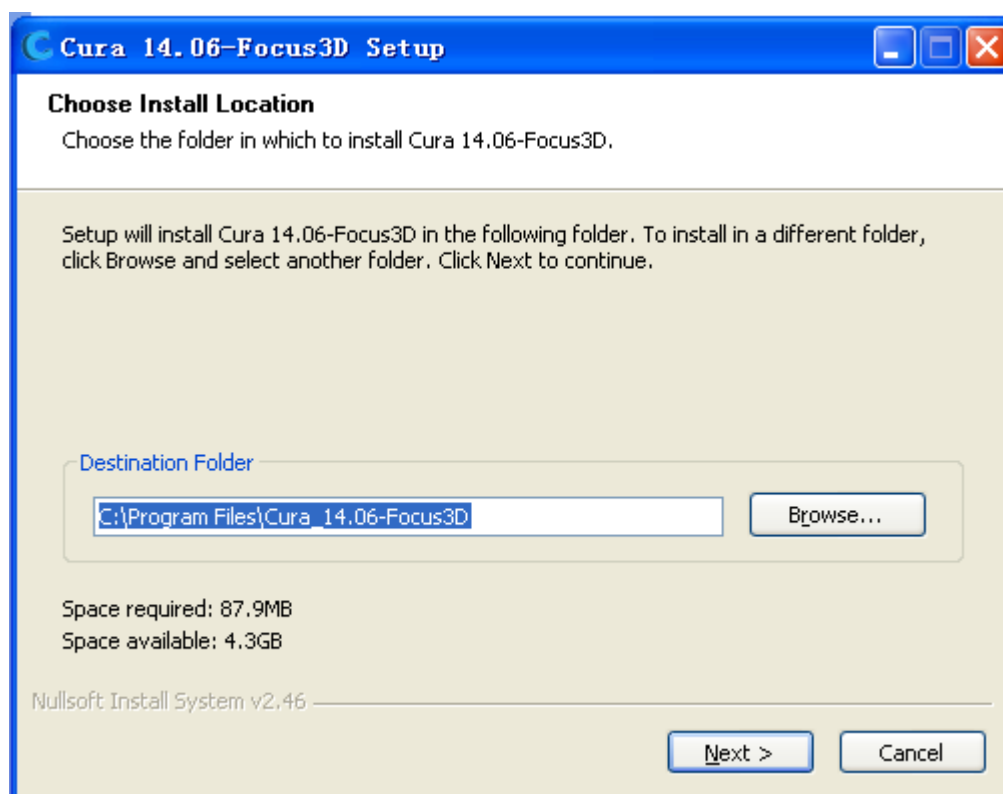
5-> Check the print platform. Please move the print head to each corner and adjust.

If the platform is not very level, you can adjust the distance between print head and the platform on all four corners by loosing or tightening the screws slightly.

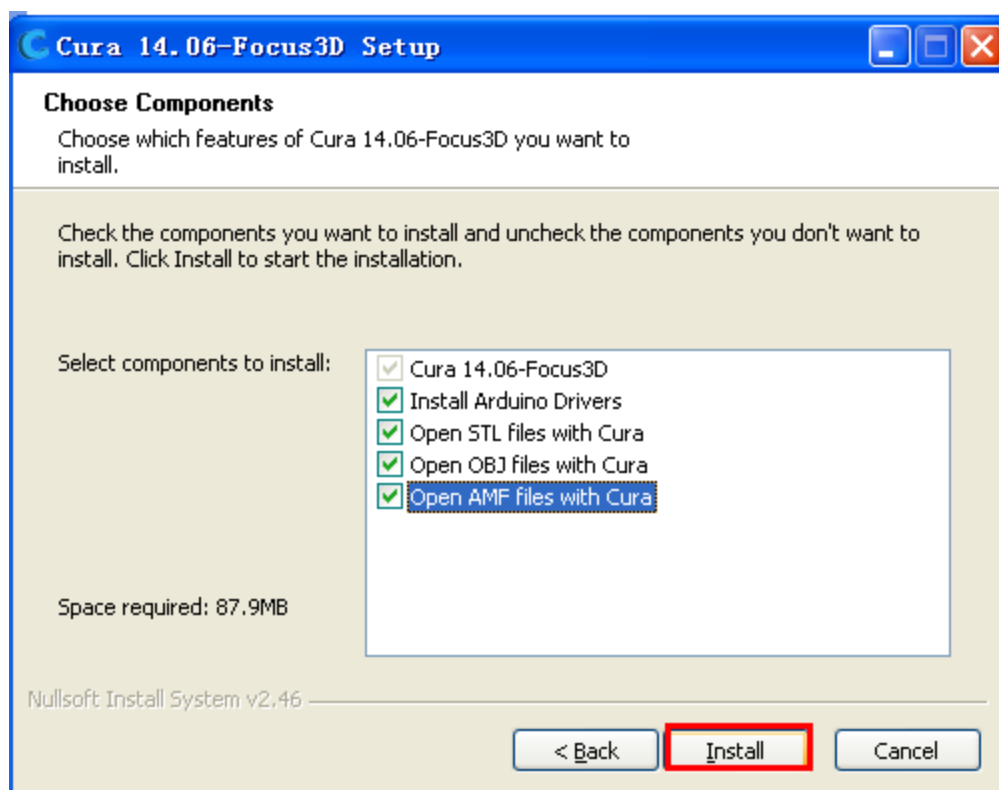
Part III: Install the Software

1-> Install Cura_14.06-Focus3D

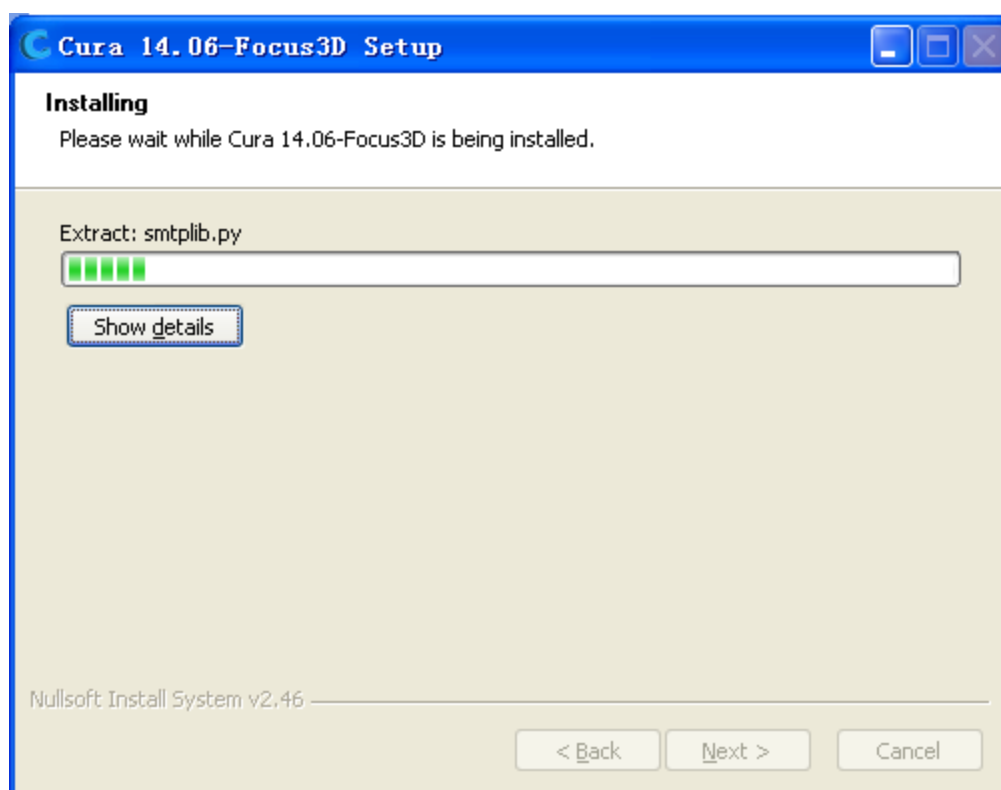
1.1> Double click “Cura_14.06-Focus3D.exe” and click “ Browse” to select the installation path and click “ next”.



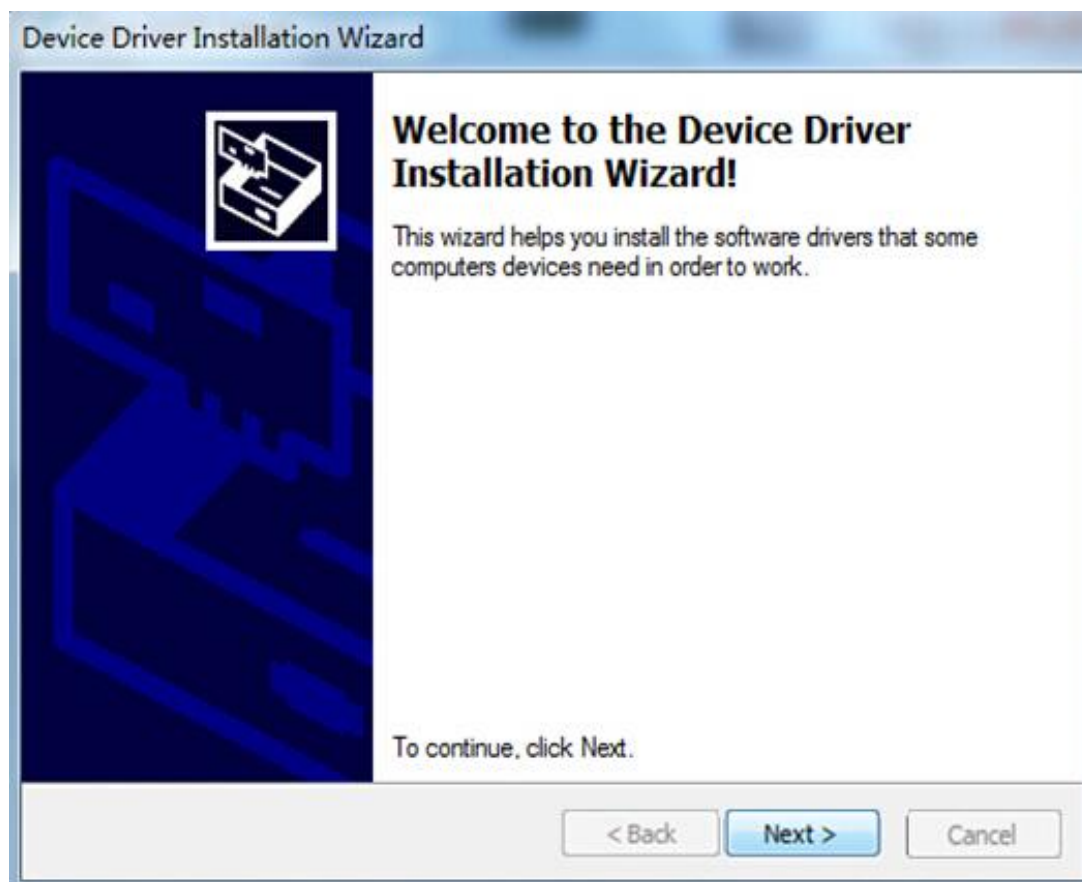
1.2> Select Focus3D Software compatible file formats you would need and click “ Install”.



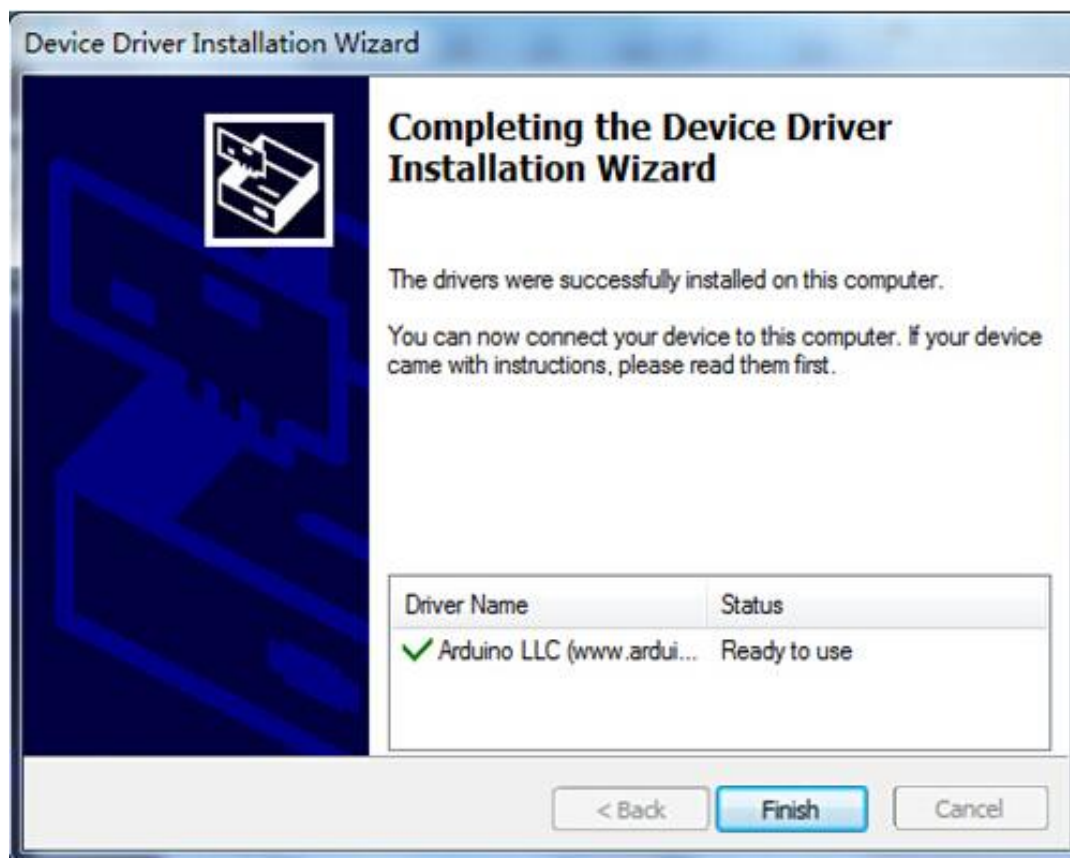
1.3> Focus3D Software Installing



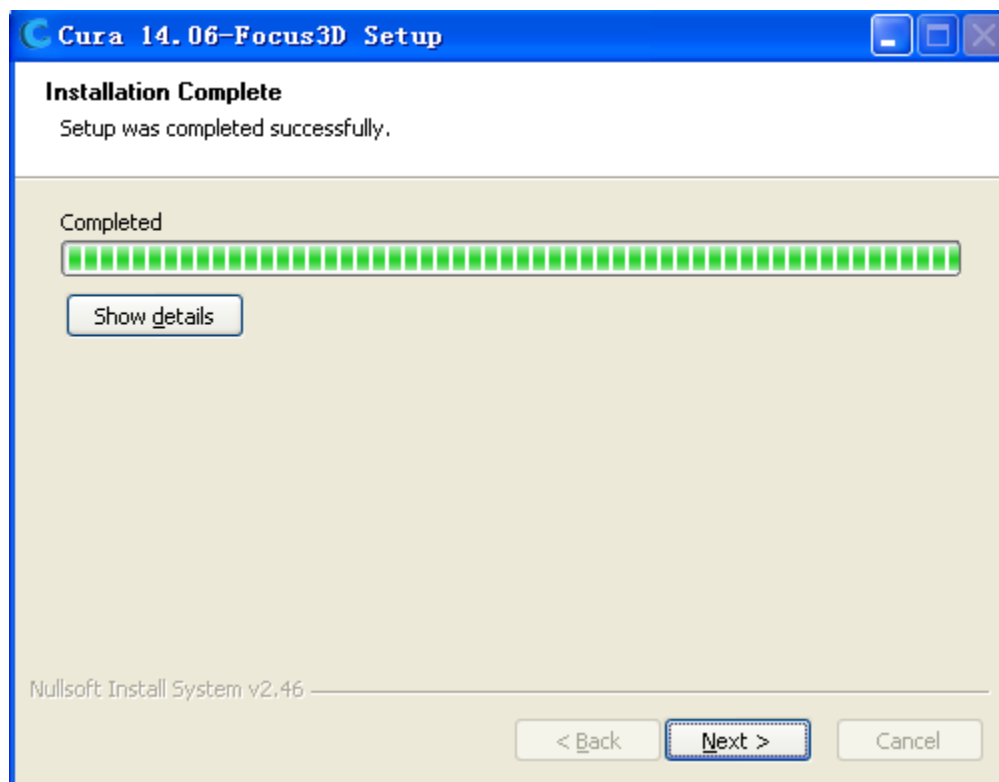
1.4> Installation finished and click “Next”.



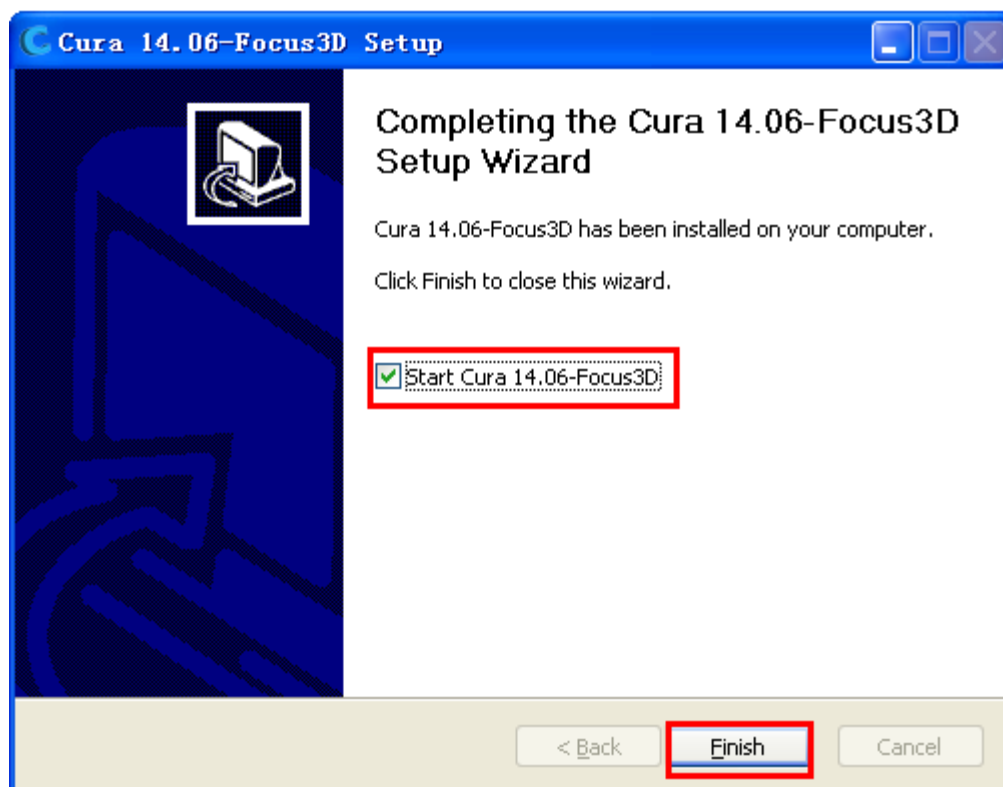
1.5> Click “Finish”



1.6> Click “Next”



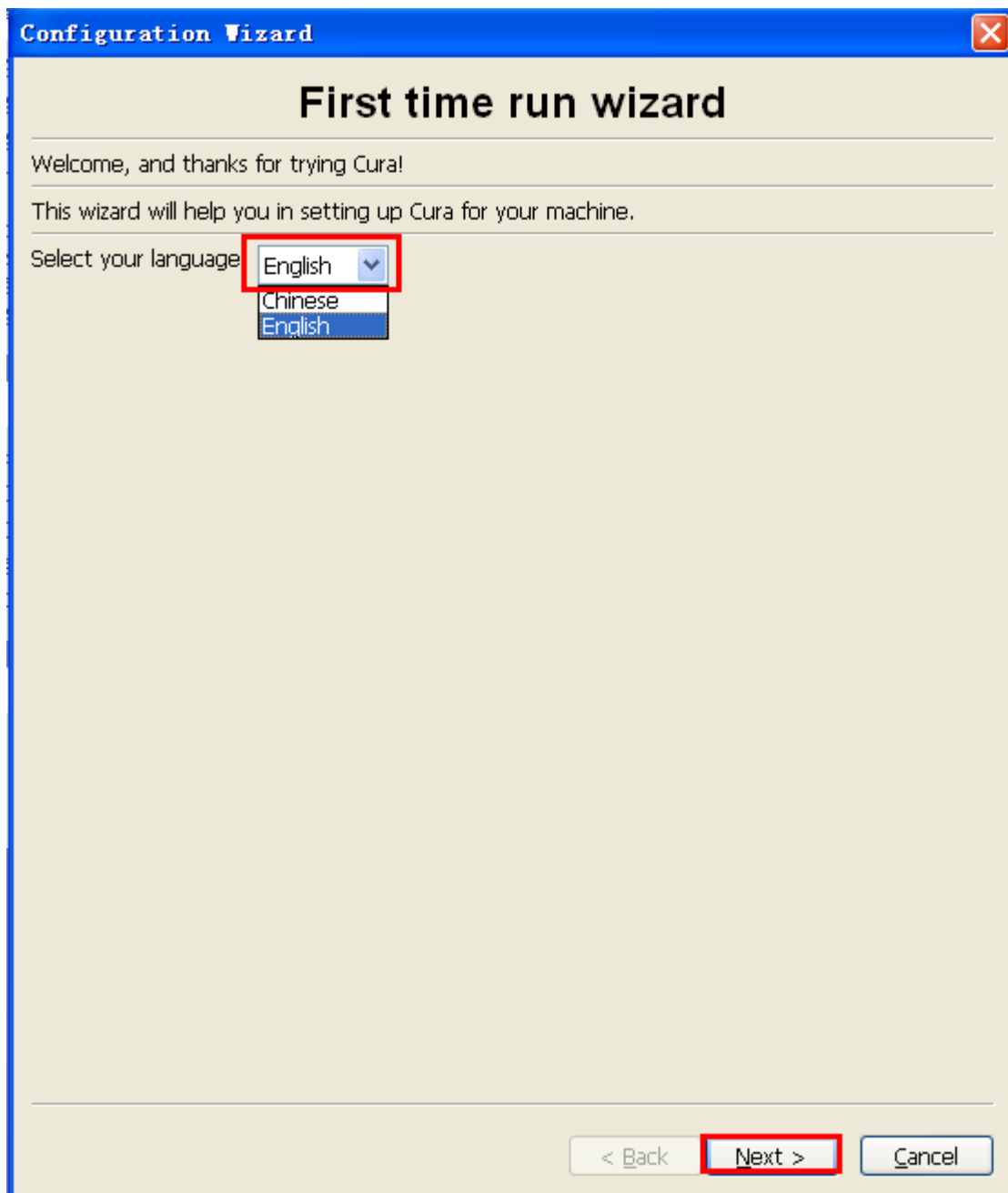
1.7> Click “Finish” to complete the software installation.



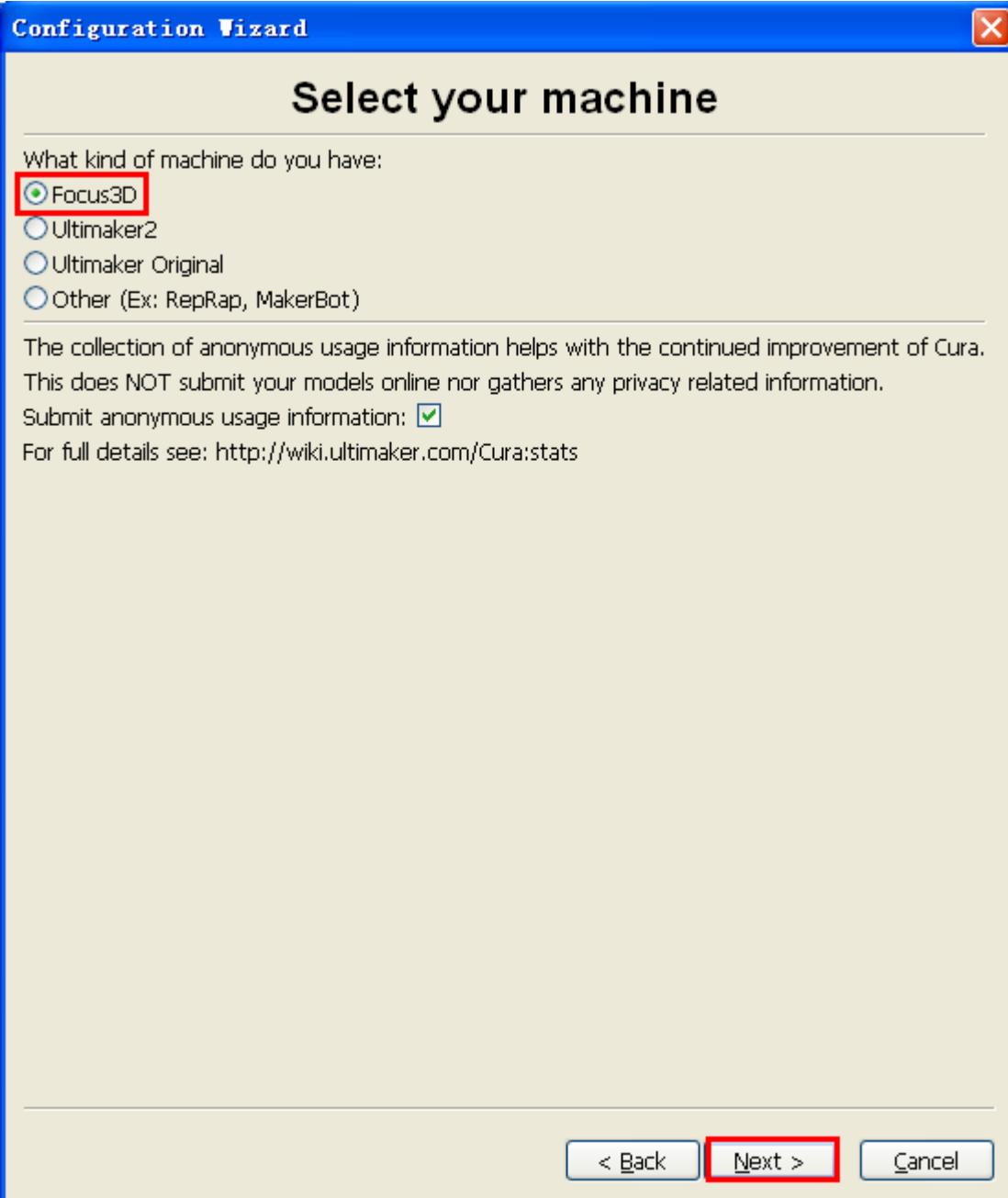
Part IV: Run the Software

1> Software Running Preparation

1.1> First Time Run Wizard and select the language.



1.2> Select Focus3D Machine Type



Configuration Wizard

Select your machine

What kind of machine do you have:

- ☒ Focus3D
- ☐ Ultimaker2
- ☐ Ultimaker Original
- ☐ Other (Ex: RepRap, MakerBot)

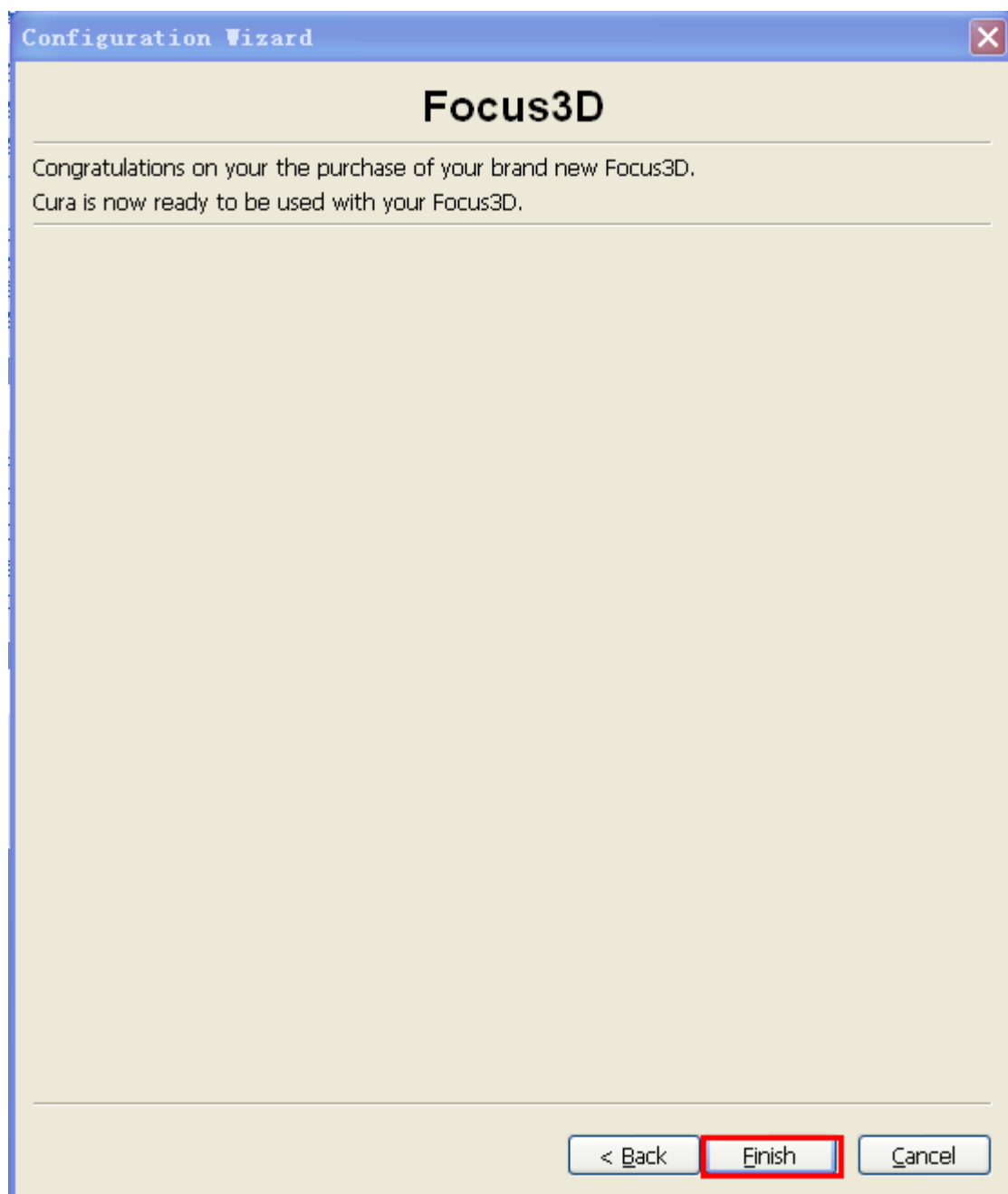
The collection of anonymous usage information helps with the continued improvement of Cura.
This does NOT submit your models online nor gathers any privacy related information.

Submit anonymous usage information: ☒

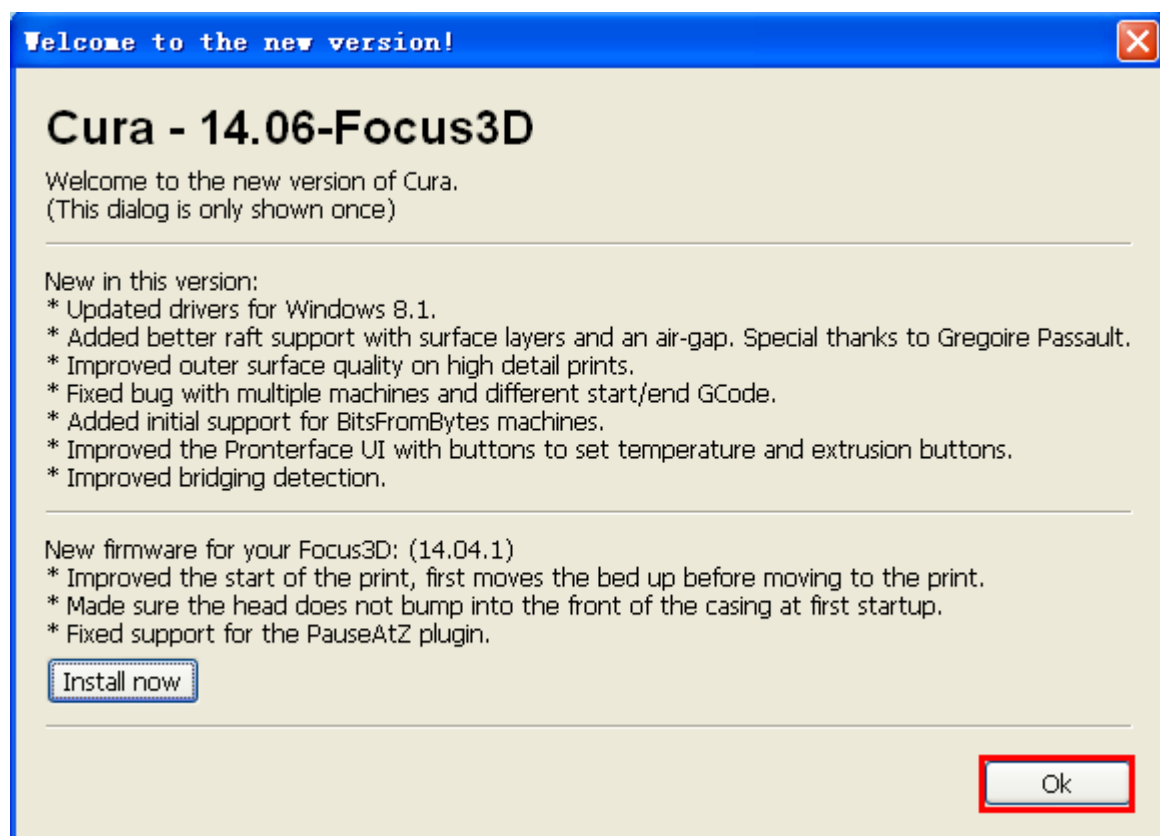
For full details see: <http://wiki.ultimaker.com/Cura:stats>

< Back **Next >** Cancel

1.3 Ready the software



1.4 Click “OK”.

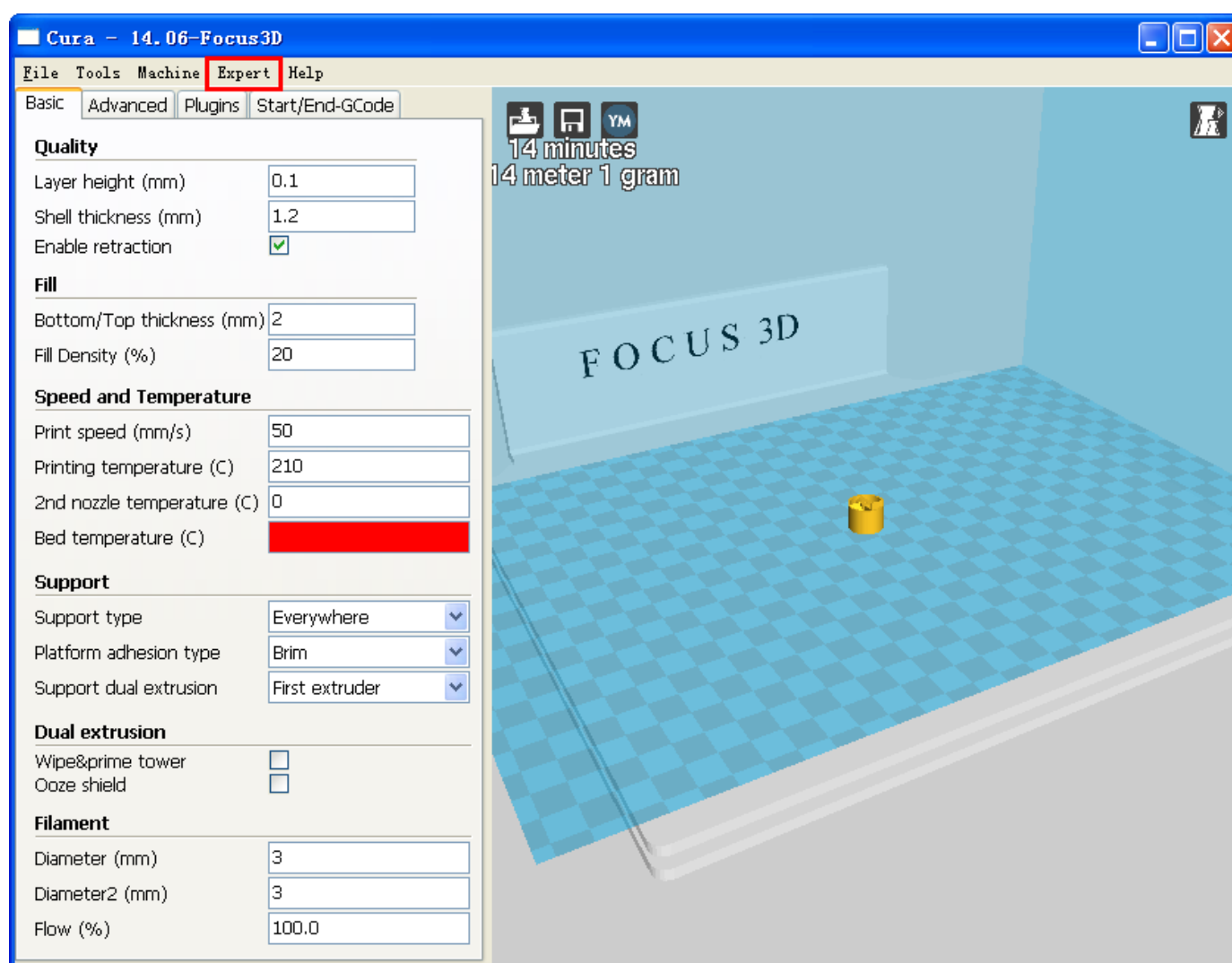


Part V: 3D Print Model Parameters Setting

1> Print Model Parameters Configuration

1.1 Switch to Full Setting

Click “Expert” and click “Switch to Full Setting”.



1.2 Focus3D Software Parameter Configuration

1.2.1. Basic Configuration

The screenshot displays the Focus3D software interface with the 'Basic' tab selected. The interface is divided into several sections: Quality, Fill, Speed and Temperature, Support, Dual extrusion, and Filament. Red arrows point from specific parameters to callout boxes explaining their function.

Section	Parameter	Value	Explanation
Quality	Layer height (mm)	0.1	The height of each layer (MAX less than 80% of the Nozzle diameter and MIN more than 0.02mm)
	Shell thickness (mm)	1.2	Retract some filament when the Nozzle is in non-printing area
Fill	Bottom/Top thickness (mm)	2	The thickness of the bottom and top (integral multiple of layer height)
	Fill Density (%)	20	The Density of the internal of the printing product
Speed and Temperature	Print speed (mm/s)	50	The speed of the traveling nozzle (normally from 50 to 100)
	Printing temperature (C)	210	The temperature of nozzle during printing (210 for PLA and 230 for ABS)
Support	2nd nozzle temperature (C)	0	Heated bed temperature. 0 for PLA, 100 for ABS
	Bed temperature (C)		
Support	Support type	Everywhere	Provide support when printing products ungrounded
	Platform adhesion type	Brim	Add thick bottom or top for the printing products
Dual extrusion	Support dual extrusion	First extruder	
	Wipe&prime tower	<input type="checkbox"/>	
Filament	Ooze shield	<input type="checkbox"/>	
	Diameter (mm)	3	
Filament	Diameter2 (mm)	3	
	Flow (%)	100.0	

Notes:

1- Layer Height will often be changed depending on the surface accuracy and the parameter is in mm (MAX less than 80% of the Nozzle diameter and MIN more than 0.02mm). Recommended 0.05mm for ultra-high-precision surface; 0.1 mm for the high-precision surface; 0.15 for good surface and 0.2mm for the general accuracy. It prints slower with lower value but with better quality . The shell thickness normally is an integer multiple of the diameter of the nozzle.

2- Choose integer multiple of the thickness for bottom and top shell thickness. Internal fill density is set to be from 0 to 100 (0 means without any fill which is for thin-walled model and 100 is for entity printing). Generally 20 percent is enough for support strength. filling the smaller ratio It can also enhance printing speed with smaller fill density.

3- Printer nozzle movement speed is generally recommended value 80mm/s. This machine can reach a maximum speed of 150mm/s, but it would lead to out of step and the print quality would be declined.

The nozzle temperature is slightly different depending on the printing material. 210C is recommended for PLA and the hot bed as 0C while 230C is recommended for ABS and hot bed as 100C.

4- Platform Options: “Brim” is to print at the bottom layer and is most recommended since it’s easy to peel. “Raft” is to print a thick base at the bottom. Print bottom support can reduce plastic shrinkage deformation. “None” means printing model directly.

5- Filament diameter can be obtained directly by measuring the filament.

6- Flow (%) is used to adjust the feeding rate

1.2.2. Advanced Configuration

The screenshot shows the 'Advanced' tab of a 3D printer configuration interface. The window is divided into several sections: Machine, Retraction, Quality, Speed, and Cool. Each section contains various settings with input fields. Red arrows point from specific settings to explanatory text boxes on the right.

Section	Setting	Value	Description
Machine	Nozzle size (mm)	0.4	The diameter of the Nozzle (0.4mm as standard)
Retraction	Speed (mm/s)	45.0	The movement speed of Feeder during retraction
	Distance (mm)	5	The length of the retracted filament each time
	Dual extrusion switch amount (mm)	16.5	
Quality	Initial layer thickness (mm)	0.3	
	Cut off object bottom (mm)	0.0	
	Dual extrusion overlap (mm)	0.15	The parameter for dual extrusion
Speed	Travel speed (mm/s)	100.0	The movement speed during non-printing
	Bottom layer speed (mm/s)	20	
	Infill speed (mm/s)	0.0	
	Outer shell speed (mm/s)	30.0	The lower the outer shell speed, the smoother or better the surface of the object. 30 is recommended
	Inner shell speed (mm/s)	50.0	The speed of printing inner parts. It can be faster.
Cool	Minimal layer time (sec)	5	Minimum time for each layer (to make sure that each layer has enough time for cooling down
	Enable cooling fan	<input checked="" type="checkbox"/>	

Notes:

1- The supplier will provide Nozzle diameter size. The standard for Focus3D Nozzle diameter is 0.4mm

2- The bottom thickness of the model, the thicker and the easier for the model to be adhesive with the platform. 0 means the thickness is the same with the other layer height.

3- Print after the model be cut off at the bottom. It’s applied to the model that does not have a flat bottom.

4- Dual nozzles imposed together which makes printed two lines overlap, This makes the lines fit together more closely printed by the two nozzles .

5- Bottom printing speed, that is nozzle moving speed of the first layer. Generally it is set lower and it allows the first layer of the model to be better adhered to the printing platform.

6- Print speed of the filling inside. 0 means the normal printing speed. The faster of the speed, the shorter of the printing time. However it will impact the printing quality.

7- The minimum time to print each layer since each layer it need sufficient time to cool down the extruded material. In this way, the value can not be set too small.

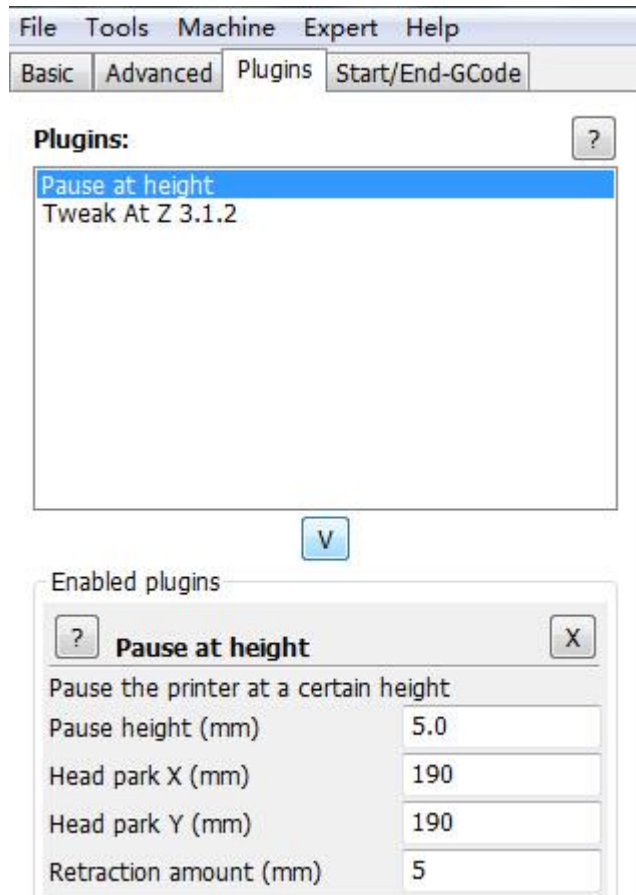
1.2.3 Expert Configuration

Expert config	
Dual extrusion	Support
Wipe&prime tower volume per layer (mm3)	Structure type
15	Grid
Retraction	Overhang angle for support (deg)
Minimum travel (mm)	80
Enable combing	Fill amount (%)
<input checked="" type="checkbox"/>	15
Minimal extrusion before retracting (mm)	Distance X/Y (mm)
0.02	0.7
Z hop when retracting (mm)	Distance Z (mm)
0.0	0.15
Skirt	Black Magic
Line count	Spiralize the outer contour
1	<input type="checkbox"/>
Start distance (mm)	Only follow mesh surface
3.0	<input type="checkbox"/>
Minimal length (mm)	Brim
150.0	Brim line amount
Cool	20
Fan full on at height (mm)	Raft
5.0	Extra margin (mm)
Fan speed min (%)	5.0
50	Line spacing (mm)
Fan speed max (%)	3.0
100	Base thickness (mm)
Minimum speed (mm/s)	0.3
10	Base line width (mm)
Cool head lift	1.0
<input type="checkbox"/>	Interface thickness (mm)
Infill	0.27
Solid infill top	Interface line width (mm)
<input checked="" type="checkbox"/>	0.4
Solid infill bottom	Airgap
<input checked="" type="checkbox"/>	0.22
Infill overlap (%)	Surface layers
8	2
	Fix horrible
	Combine everything (Type-A)
	<input checked="" type="checkbox"/>
	Combine everything (Type-B)
	<input type="checkbox"/>
	Keep open faces
	<input type="checkbox"/>
	Extensive stitching
	<input type="checkbox"/>
	Ok

1.2.4 Plugins Settings

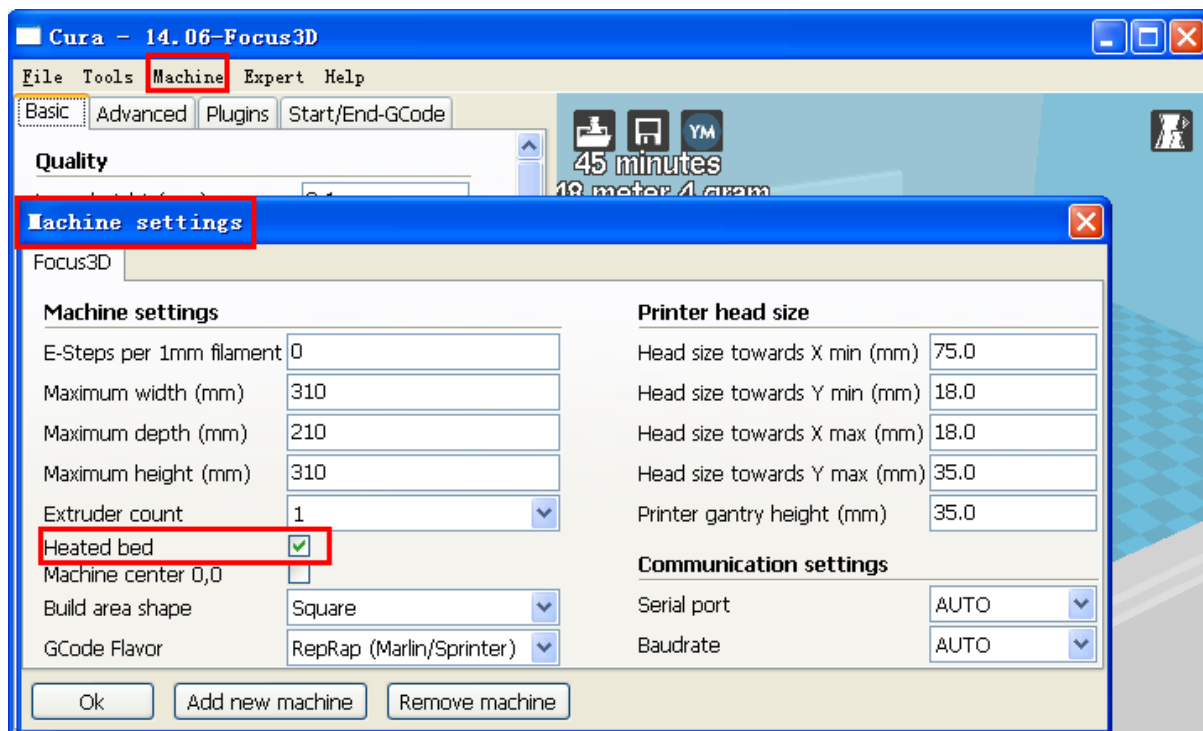
You can pause the printing at specific height throughout this setting.

Click “V” and set the pause position at a certain height and also set the length of retraction.



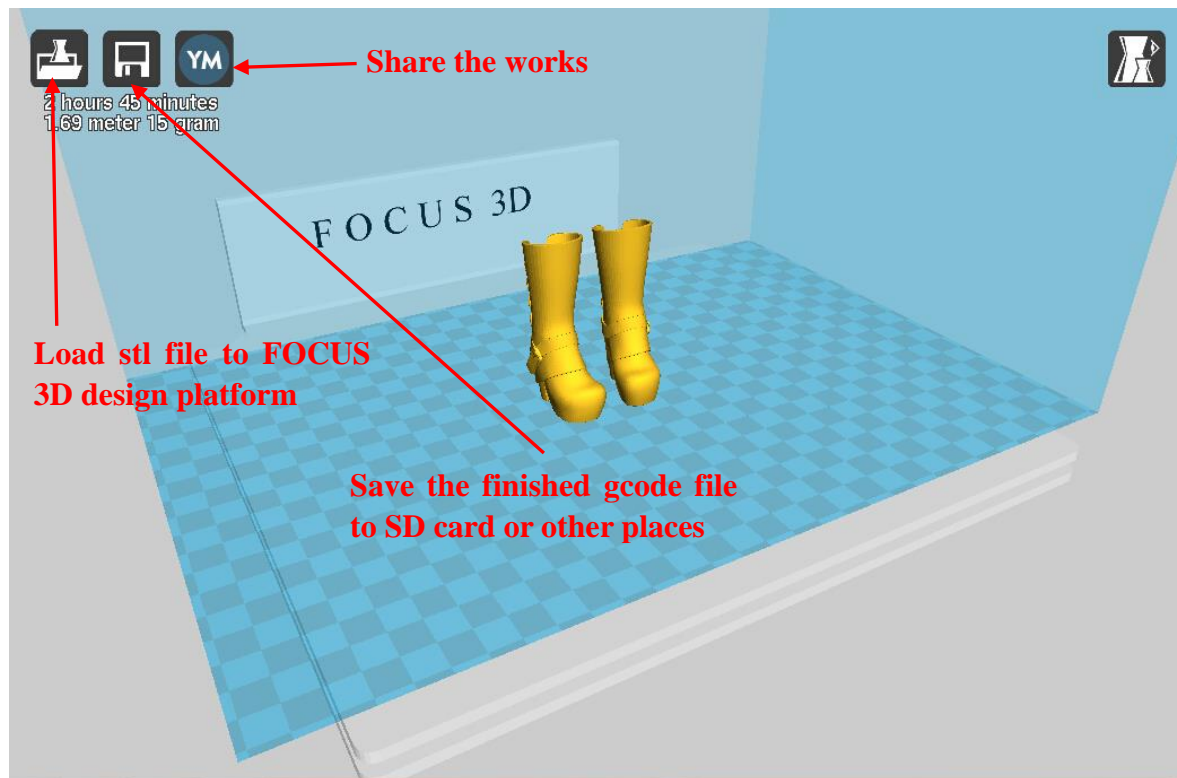
1.2.5 Machine Settings

Click “Machine”, and select” Machine Settings”. **If you print ABS, please add “Heated bed”.**

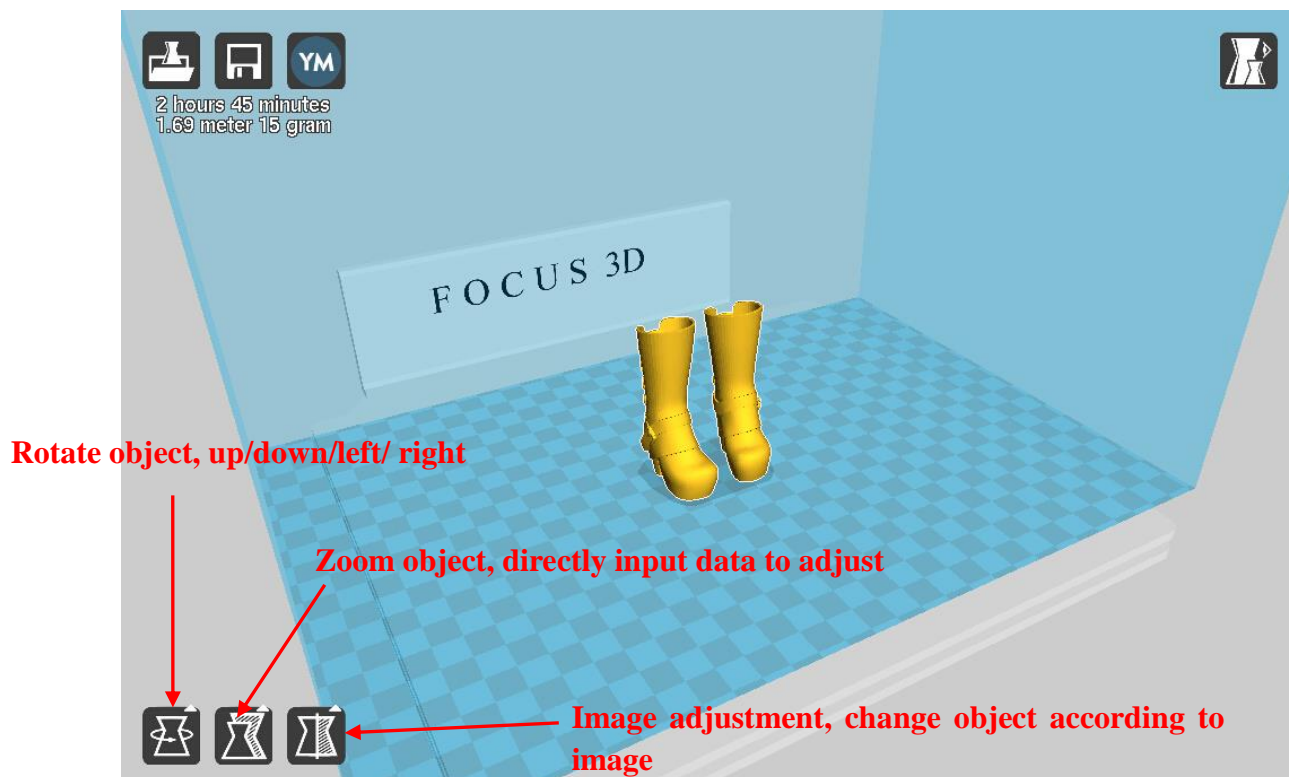


Part VI: Start to Print

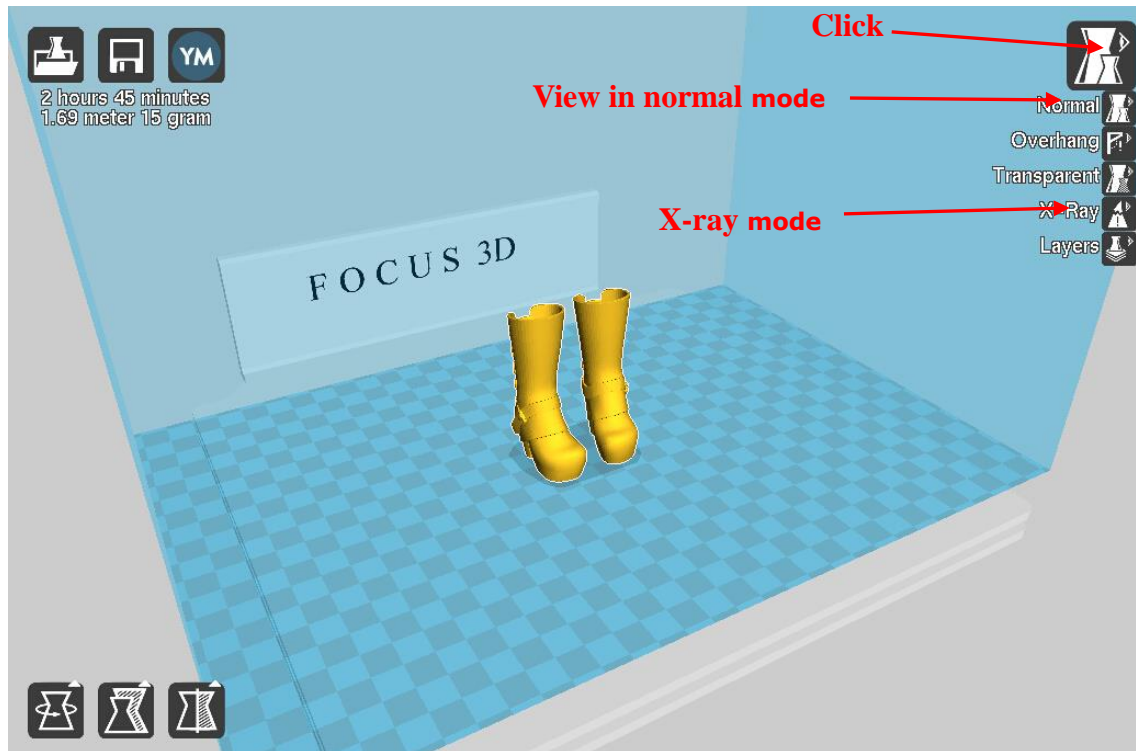
1-> Load the stl file and adjust the model size



Rotate, Zoom, Mirror to adjust the file

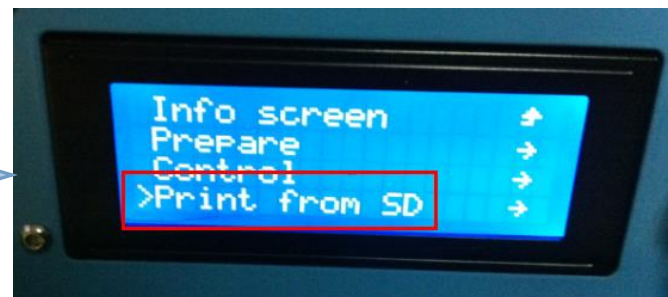


File View Pattern

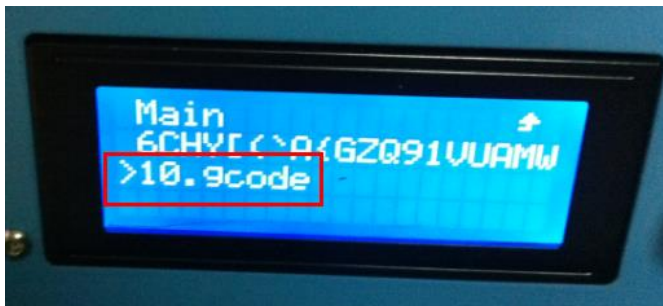


2-> Save the gcode file to a SD card. Insert SD card to 3D printer and select the file to print.

2.1 Insert the SD card



2.2 Select the file to print



The 3D printer will start to print.

Notes:

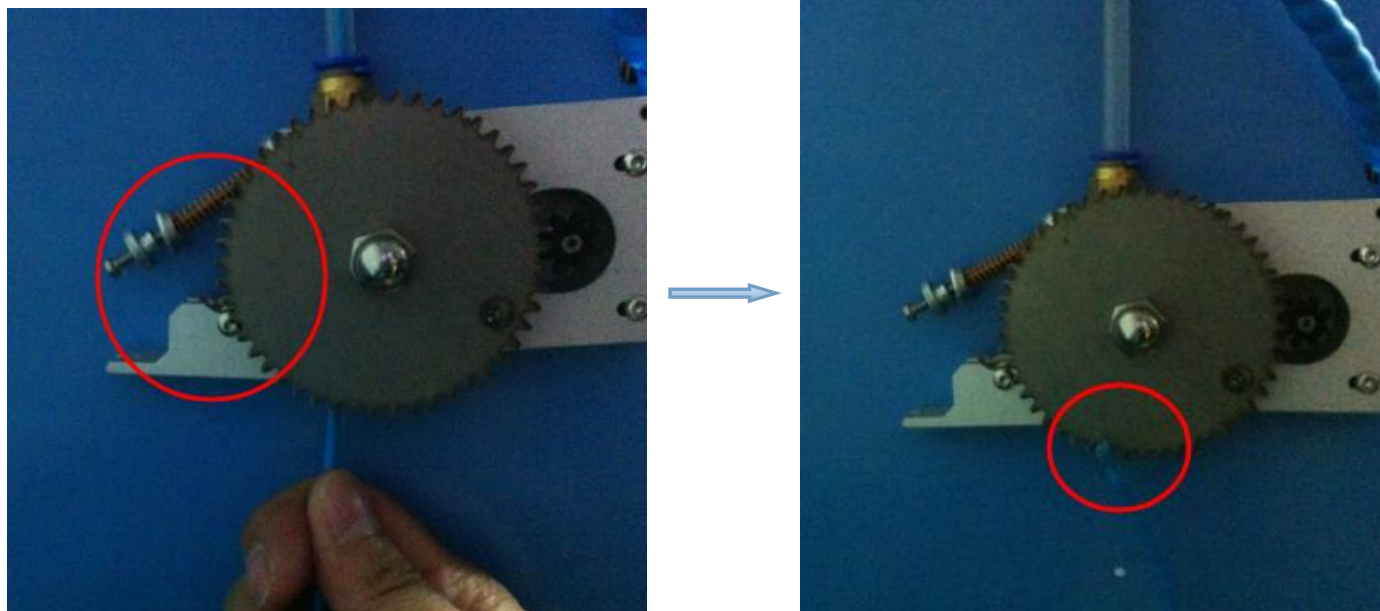
When parameter settings are completed and the model position is adjusted then Focus3D software will automatically generate Gcode print documents. If it's generated correctly, then we will find the file in the directory named as .Gcode.

Part VII: Change the Filament

1> Prepare



2> Loose the gear, Pull out the filament



3> Fill the new filament into the extruder.

Thank you for purchasing our 3D printers. If you have any more technical problem, please email us with your questions. We will provide you with timely help.



044 237-66-06